Superelement Analysis of Tile-Reinforced Composite Armor

Carlos G. Dávila

Army Research Laboratory, Vehicle Technology Center
NASA Langley Research Center, MS 190
Hampton, VA 23681

Abstract

Superelements can greatly improve the computational efficiency of analyses of tile-reinforced structures such as the hull of the Composite Armored Vehicle. By taking advantage of the periodicity in this type of construction, superelements can be used to simplify the task of modeling, to virtually eliminate the time required to assemble the stiffness matrices, and to reduce significantly the analysis solution time. Furthermore, superelements are fully transferable between analyses and analysts, so that they provide a consistent method to share information and reduce duplication. This paper describes a methodology that was developed to model and analyze large upper hull components of the Composite Armored Vehicle. The analyses are based on two types of superelement models. The first type is based on element-layering, which consists of modeling a laminate by using several layers of shell elements constrained together with compatibility equations. Element layering is used to ensure the proper transverse shear deformation in the laminate rubber layer. The second type of model uses three-dimensional elements. Since no graphical preprocessor currently supports superelements, a special technique based on masterelements was developed. Masterelements are representations of superelements that are used in conjunction with a custom translator to write the superelement connectivities as input decks for ABAQUS.

Introduction

The Composite Armored Vehicle (CAV) upper hull consists of layers of glass-epoxy materials, rubber isolation mats, and ceramic armor tiles. The CAV achieves its ballistic damage resistance from the interaction of these materials which have very different stiffnesses. The ceramic tiles, by far the stiffest component in the laminate, support the largest portion of the load. However, the tiles are only 4 inches long and the load must flow across the narrow adhesive gaps between the tiles, as illustrated in Fig. 1. This type of response can be simulated by using detailed three-dimensional finite element models [Dávila and Smith, 1996]. However, the small size of the details that must be included for adequate response predictions precludes the use of such models for large components.

One technique that has been successfully demonstrated on large structural CAV components is based on homogenization theory [Dávila and Chen, 1996]. This technique smears out the discontinuities in material properties, and is able to simulate accurately the global response of the structure. However, homogenization theory is not useful for the prediction of strength, since the stresses are only average values for the different constituents. This averaging procedure also prevents the prediction of strength by the use of the global-local method, in which the displacements from a global model are applied to a detailed local model. Finally, homogenization is not applicable to sections with small characteristic length that are often encountered in the CAV.

This paper proposes a technique based on substructuring that overcomes many of the computational difficulties associated with detailed or homogenized finite element models. The efficiency of the method is based on the observation that tile-reinforced structures are composed of repeating units. Substructuring makes use of this periodicity to: i) simplify the task of modeling; ii) eliminate the duplication associated with the formation of identical stiffnesses more than once; and iii) reduce the number of variables with static condensation.

The remainder of this paper is presented as follows. First, the theory, advantages and limitations of superelement analysis are discussed. Then, a technique to generate superelements and superelement models is described. Finally, some sample analysis results are presented.

Superelements: Theory, Advantages and Limitations

The basic concept of substructuring consists of separating the model into parts, and eliminating all degrees of freedom except for those needed to connect the parts to the rest of the model. A reduced part appears in the model as a “superelement:” a collection of nodes and degrees of freedom connected by the superelement’s stiffness and mass matrices. The reduced stiffness matrix is easily derived by partitioning the matrix of the substructure into the terms that multiply the retained degrees of freedom, u_r, and those that multiply the condensed (eliminated) degrees of freedom, u_c, as follows.

$$\begin{bmatrix} K_{cc} & K_{cr} \\ K_{rc} & K_{rr} \end{bmatrix} \begin{Bmatrix} u_c \\ u_r \end{Bmatrix} = \begin{Bmatrix} P_c \\ P_r \end{Bmatrix} \tag{1}$$

From the first equation, extract u_c in terms of the retained degrees of freedom, u_r

$$u_c = K_{cc}^{-1}\left(P_c - K_{cr}u_r\right) \tag{2}$$

and after substituting u_c into the second line of Eq. 1 and rearranging, the condensed stiffness K^* is

$$K^* u_r = P^* \quad \text{where} \quad \begin{cases} K^* = K_{rr} - K_{rc}K_{cc}^{-1}K_{cr} \\ P^* = P_r - K_{rc}K_{cc}^{-1}P_c \end{cases} \tag{3}$$

Substructuring introduces no additional approximation in static analysis; the superelement is an exact representation of the linear, static behavior of its members. However, the formation of the matrix K^* and the reduced load vector P^* are obtained at a computational cost equivalent to that of performing a Gaussian elimination of the degrees of freedom u_c from the full model's assembled stiffness matrix. In other words, no benefit is achieved from static condensation unless the superelement can be re-utilized in subsequent analyses or substructures.

Once the system's solution has been obtained, it is possible to recover the internal response of the superelement, provided that all terms in Eq. 2 are saved in the database. [ABAQUS Theory Manual, 1995].

Superelements offer a number of advantages, some directly related to CAV analyses, and others related to general aspects of the analysis. Some of these advantages are:

- System matrices (stiffness, mass) are small as a result of condensation. Only the superelement's retained degrees of freedom are used in the analysis.
- Duplication in the formation of the substructure's stiffness is eliminated when the same superelement is re-used.
- Superelement library files allow analysts to share superelements. In large design projects, large groups of engineers must often conduct analyses using the same substructures. Superelement library files provide a straight forward and simple way of sharing structural information.
- The task of modeling can be simplified by including intricate details inside the superelement. For instance, the small adhesive gap around a ceramic tile does not need to be modeled when using superelements.

The following list itemizes the limitations and penalties imposed by the use of superelements

- Superelements are linear or a linear perturbation about a nonlinear state. In a nonlinear analysis, the stiffness is a function of the accumulated displacements, which vary from element to element. Therefore, even identical superelements no longer have the same stiffness matrix.
- Dynamic solution is approximate: the static modes obtained by a reduced set of retained nodes may not fully represent the dynamic response. Methods that are used to improve the accuracy of superelements in dynamic analyses include Guyan reduction and mode augmentation procedures [ABAQUS User's Manual, 1996]. Dynamic solutions are not considered in this paper.
- The large number of degrees of freedom of a single superelement causes a large bandwidth, and highly populated matrices that can mitigate some of the computational advantages of superelements. Judicious selection of the size and number of superelements can increase computational performance.
- There is no pre-processor support. This issue is the most significant limitation of superelements. A custom PATRAN-to-ABAQUS translator was developed for the present study that partially circumvents this difficulty. However, the authors expect that general-purpose graphical pre-processors such as PATRAN [PATRAN, 1997] and SDRC/Ideas [Ideas, 1997] will possess superelement capabilities in the near future.

Superelement Models and Analyses of the CAV Upper Hull

Superelements are used in ABAQUS much the same way as any other element. This procedure requires that the connectivity of the superelement is specified in the sequence in which the superelement was defined. Unfortunately, there is no graphical pre-processor that currently supports superelements, and establishing the superelement connectivities for a complex model without pre-processor support is impractical. The following subsections describe the procedure developed to create superelements and superelement models using PATRAN. Then, a benchmark test comparing the performance of a standard analysis and a superelement analysis is described. Finally, examples of analysis results postprocessed with the graphical post-processor ABAQUS/Post are provided for illustration.

1) Generating a Superelement with PATRAN

The basic repeating unit cell in a tile-reinforced panel can consist of any pattern created by the tiles: these patterns can range from a quarter of a tile to groups of several tiles. The schematic representation shown in Fig. 2 consists of a typical full-tile substructure consisting of one tile, half of the gap surrounding it, and the composite and rubber layers above and below the tile. This mesh is designed to have even node spacing around the circumference of the cell to allow connections between staggered tiles. In the present analyses, the basic unit cell was chosen to be a quarter of a tile. Half-tile and full-tile superelements were also created by combining two or four quarter-tile superelements.

A superelement is defined in ABAQUS much like any typical finite element model. It consists of items such as node locations, element connectivities, material properties, and, optionally, constraint equations or boundary conditions, all of which are easily created using PATRAN. In addition, it is necessary to identify and sort the retained nodes of the substructure. The ordered list of retained nodes defines the nodal connectivity of a superelement. The ordering of the list is arbitrary, but it must be the same as in the models that use the superelement. In relatively complex problems such as the four-layer superelement example shown in Fig. 3, the ordering of the nodes is performed using a small FORTRAN program.

2) Creating Superelement Models in PATRAN using Masterelements

Graphical pre-processors such as MSC/PATRAN and SDRC/Ideas are powerful feature-laden tools that have become indispensable for generating any practical finite element model. Unfortunately, no pre-processor currently supports superelements. In particular, to create a superelement model, a pre-processor must at least be able to i) represent a superelement; ii) clone a superelement with specified translations and rotations to position the superelement in the model; and iii) export the superelements into the appropriate (ABAQUS) finite element input decks.

The absence of these necessary features in PATRAN was overcome with the use of masterelements and a custom PATRAN-to-ABAQUS translator. Masterelements are nothing more than triangular and bar elements arranged to simulate a superelement. The triangular elements connect all the retained nodes along the periphery of the superelement, as illustrated in Fig. 4. The third node of all triangles is at the center of the superelement. The PAT_ABA translator determines the connectivity of a superelement by grouping and sorting all the nodes belonging to the same superelement. In addition, two types of bar elements are used, and these are identified by their PATRAN property number. Orientation bars indicate the position and spatial orientation of the superelement. The corresponding translation and rotation of the superelement is written in the *SUPER PROPERTY card record [ABAQUS User's Manual, 1996]. The second type of bar is used in element-layered superelements to connect the layers together. The retained nodes are sorted by layer, and within a layer, by their counter-clockwise position starting from the location of the orientation bar.

Modeling superelements with masterelements is unlike the typical meshing of finite elements. Instead of defining a finite element mesh over a geometric entity such as a line, a surface, or a volume, using masterelements requires "paving" the geometry with masterelements. The user starts with a database containing several different masterelements, such as a full tile, a half tile and a quarter tile. Each masterelement is contained within its own *group*, so that it can be selectively displayed and manipulated. In PATRAN, *groups* contain parts of a model that can be selectively posted to or unposted from the viewport. Using groups, the masterelements are cloned into their proper location, one at a time, until the entire part has been paved. The original masterelement groups can then be deleted, as they are no longer needed.

The most efficient method of working with superelements is to pre-define libraries of compatible elements that can be reused in similar analyses. For instance, the present study is based on three libraries of elements: *L4.N3* (Fig. 5), *L4.N8*, and *3D.N3* (Fig. 6) [Dávila and Baker, 1997]. The letter *L* indicates a superelement based on element-layering, followed by the number of layers used. The notation *3D* indicates a superelement based on solid continuum elements. The number following the letter *N* indicates the level of mesh refinement. The lowest level of mesh refinement, N3, uses three elements along the side of one quarter of the tile, while N8 uses eight. All three libraries are composed of three superelements. The most basic superelement is the quarter-tile. Half-tile and full-tile superelements were built using two or four quarter-tile superelements, respectively. For the user's modeling convenience, PATRAN templates are provided for each of the superelement libraries.

The element-layered libraries L4.N3 and L4.N8 are among the most computationally effective meshes that can be used to predict the response of tile-reinforced laminates. They derive their efficiency from the use of multiple layers of shear deformable 4-node quadrilateral shell elements (S4R) tied together with multi-point constraints. While these constraints can be difficult to write, these constraints are internal to the superelements. Therefore, the constraints are totally invisible to the user of pre-defined superelement libraries. However, element-layered models can be significantly more difficult to use than three-dimensional models. First, in problems where conventional elements must also be used, the user has to write the constraints. Also, care must be exercised when applying displacement boundary conditions; out-of-plane (z-direction) and rotation constraints can only be applied to the lower layer. The corresponding degrees of freedom in the upper layers have been eliminated through the constraints.

3) Element-Layered Benchmark Analysis

The comparative analyses shown in Fig. 7 were performed using standard elements and using superelements. Both models are based on element-layering and their mesh refinement is the same. The example was selected based on the computational cost of the standard model. The mesh refinement is relatively low, with elements in the gap region possessing aspect ratios as high as 50-to-1. A more refined mesh, a larger plate, or a three-dimensional model would have resulted in an impractically large conventional model. The standard model uses 15,885 multi-point constraint equations (MPCs). All the MPCs in the superelement model are internal to the superelement and, therefore, invisible to the user.

The results of the comparative study are shown in Table 1. As expected, the results for both analyses are identical. However, the solution was obtained in 1/12th of the time by using superelements. The reduction in solution time is not only the result of a reduced number of equations, but is also due to the reduction in the CPU time required to form and assemble the problem's stiffness matrix. When using superelements, the stiffness matrices are available in libraries, and do not need to be formed.

Another large difference between the two analyses is the total file size used by ABAQUS. The standard analysis uses 3,500 times more disk space because the problem is too big to fit into memory. Therefore ABAQUS needs to perform part of the solution using a swap file.

4) CAV Analyses and Postprocessing Examples

In the benchmark problem shown in Fig. 7, the entire model was paved with superelements. In the more complex problem shown in Fig. 8, however, conventional elements must be used in areas where boundary conditions are not aligned with the retained nodes. This problem was analyzed using three libraries of superelements: *L4.N3, L4.N8*, and *3D.N3*.

A comparison of the results for the three libraries is shown in Table 2. The applied load is 30,000 lbs. The results indicate that the model using L4.N3 is only 1.5% stiffer than the model with L4.N8, in spite of a considerably less refined mesh. The deflections obtained from 3D.N3 are 25% larger, due more to differences in modeling technique and properties, than to a more converged mesh.

The ABAQUS Standard license includes a graphical post-processor called ABAQUS/Post. Post is a command-driven tool that provides graphical display of ABAQUS models and results. While the user interface in Post does not have all the functionality of general-purpose pre- and post-processors, it does support all the options available in ABAQUS. All results are organized within nested multilevel substructures. The user advances from one superelement level to another using the keywords `*SUPER PATH, ENTER ELEMENT=n` and `*SUPER PATH, LEAVE`. Once inside the desired substructure, the user directs the nodal or element data recovery desired in the same manner as any other ABAQUS analysis. To illustrate the usage of Post, the results of a model based on superelement library L4.N8 are shown in Fig. 9. For illustration, surface strains are plotted only on selected superelements. The outlines of other superelements can be identified from the patterns formed by their retained nodes.

ABAQUS/Post also works well with models using superelements based on solid elements. The results shown in Fig. 10 correspond to an analysis using library 3D.N3. The figure consists of a top level (level-0) deformation plot superposed on lower-level plots of the strains and deformation inside the first two superelements.

Concluding Remarks

The results contained herein demonstrate that superelements are a practical and computationally effective tool for the analysis of tile-reinforced structures. With superelements, it is possible to use a desktop workstation to analyze large CAV-type structural components. In particular, it was found that ABAQUS is particularly well suited for these types of analyses, since it is simple to develop efficient models based on pre-defined libraries of superelements. Although no pre-processor yet provides support for ABAQUS superelements, a relatively simple modeling technique was developed to simulate superelements in PATRAN. This technique, based on masterelements composed of triangles and bars, is used in conjunction with a custom PATRAN-to-ABAQUS translator to write the input deck for ABAQUS superelements. For post-processing, it was found that ABAQUS/Post fully supports all superelement options, and it provides a more than adequate tool for processing results.

References

Anon., ABAQUS Theory Manual, Version 5.5, Pawtucket, RI, 1995, pp. 2.13.1-2.

Anon., ABAQUS User's Manual, Version 5.6, Pawtucket, RI, 1996.

Anon., MSC PATRAN V7.5 User's Manual, Los Angeles, CA, 1997.

Anon., SDRC Ideas, Master Series Version 5, User's Documentation, Milford, OH, 1997.

Dávila, C., Baker, D., and Taylor, S., "Superelement Analysis of Tile-Reinforced Composite Armor," VTC NR 97-04, Army Research Laboratory, NASA Langley Research Center, Hampton, VA, August 1997.

Dávila, C., Smith, C., and Lumban-Tobing, F., "Analysis of Thick Sandwich Shells with Embedded Ceramic Tiles," paper presented at the DoD/NASA Conference on Composites and Structures, Fort Worth, TX, August 1996.

Dávila, C., and Chen, Tzi-Kang, "CAV Tile Layer Homogenization and Parametric Study," VSD NR 96-03, Army Research Laboratory, NASA Langley Research Center, Hampton, VA, July 1996.

Table 1. Comparison of analysis using conventional elements and superelements

	Standard	**Superelement**
Max. deflection, in.	0.585	0.585
number of nodes	6,068	1,664
number of equations	36,408	6,135
RMS. dof wavefront	932	813
Total File size, MB	354	0.1
CPU model definition, sec.	374	14
solution, sec.	303	43
TOTAL CPU, sec.	**677**	**57**

Table 2. Comparison of results: plate subjected to three-point bending

	L4.N3 Library	**L4.N8 Library**	**3D.N3 Library**
degrees of freedom	8,597	41,827	16,973
RMS. dof wavefront	2,217	6,198	2,922
max. deflection, in.	-.782	-.794	-1.06
CPU time, sec.	280	1,756	1,093

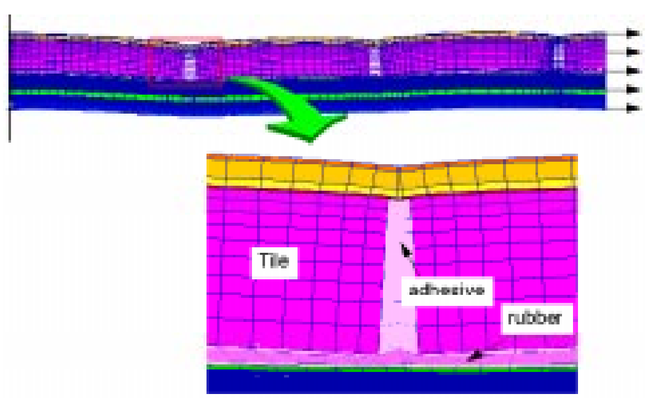


Figure 1. Plane strain analysis of CAV beam loaded in tension. High compliance of the small adhesive bondline results in a relatively large opening of the gap.

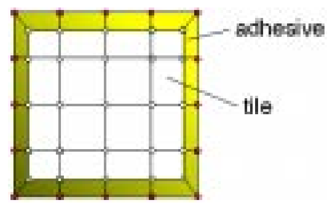


Figure 2. Schematic of substructure comprising a tile and half of the adhesive gap surrounding it. The retained nodes are uniformly spaced to allow connections between staggered tiles

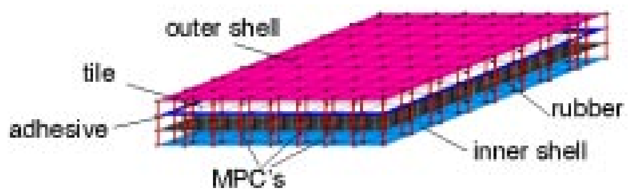


Figure 3. Superelement based on element-layering. The four material layers are tied together using Multi-Point Constraint equations.

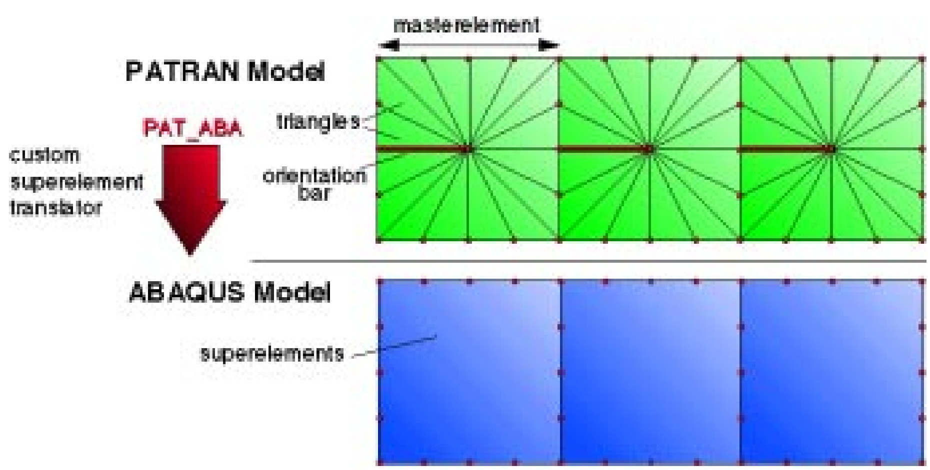


Figure 4. Triangles and bars are used to represent and clone superelements using PATRAN. PAT_ABA sorts the nodes and writes the corresponding ABAQUS input deck.

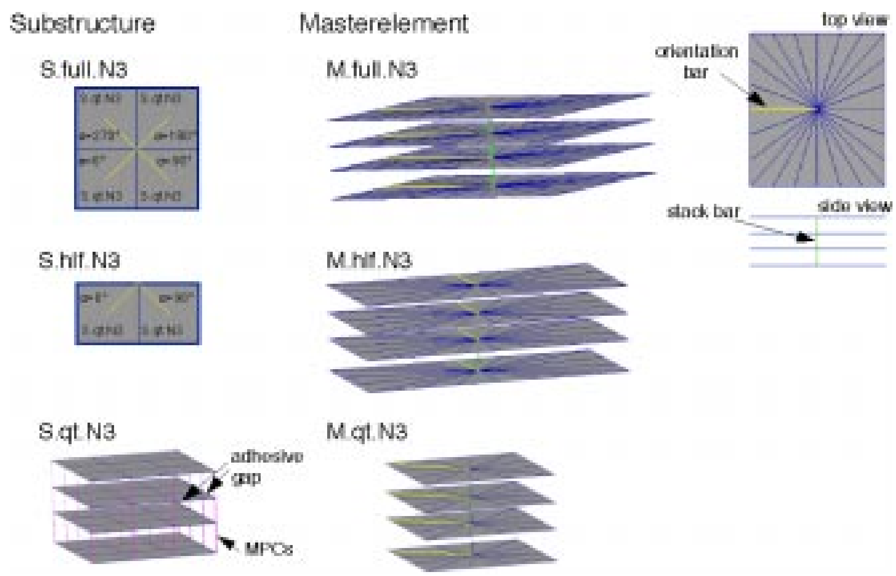


Figure 5. Superelement library L4.N3 is composed of quarter-tile, half-tile, and full-tile superelements. The corresponding PATRAN template has masterelements corresponding to each of these three superelements.

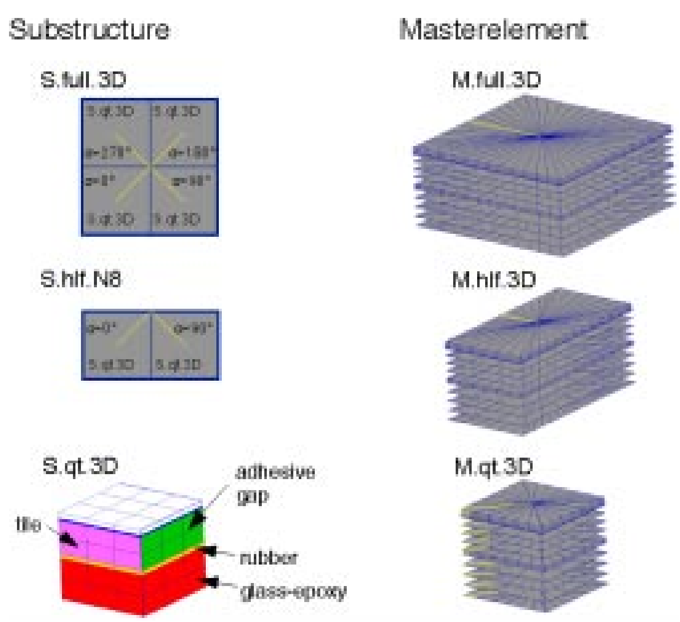


Figure 6. Superelement library 3D.N3 uses three-dimensional elements. The masterelements, however, still use triangles to represent the superelements in PATRAN.

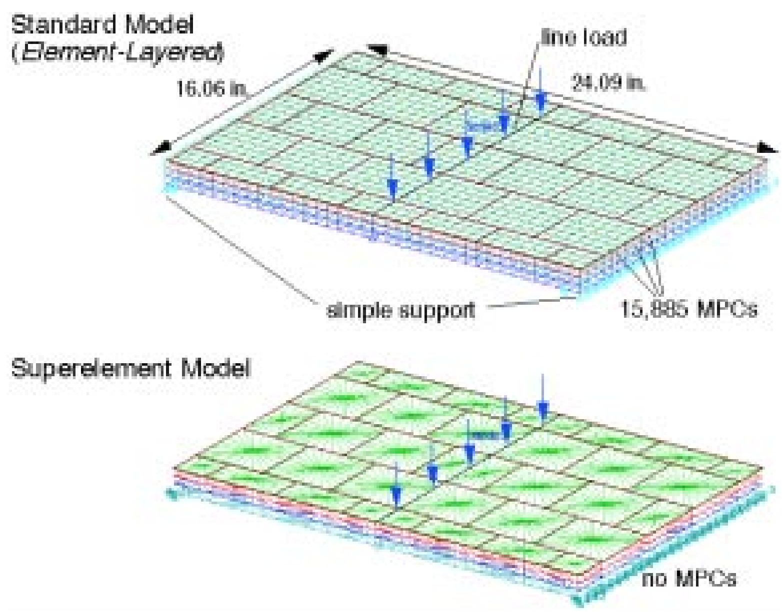
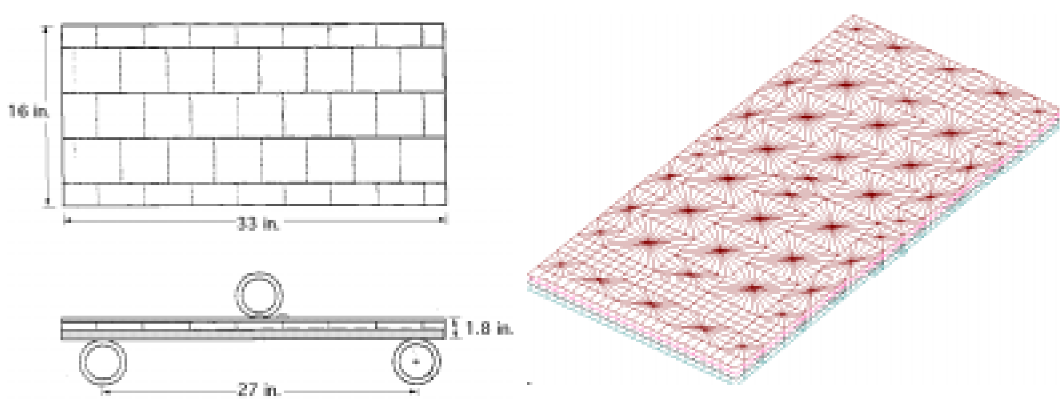


Figure 7. Comparative analyses of CAV-type composite armored plate subjected to three-point bending.



a) test setup

b) finite element model in PATRAN with masterelements and conventional elements.

Figure 8. Test setup and finite element model for plate subjected to three-point bending.

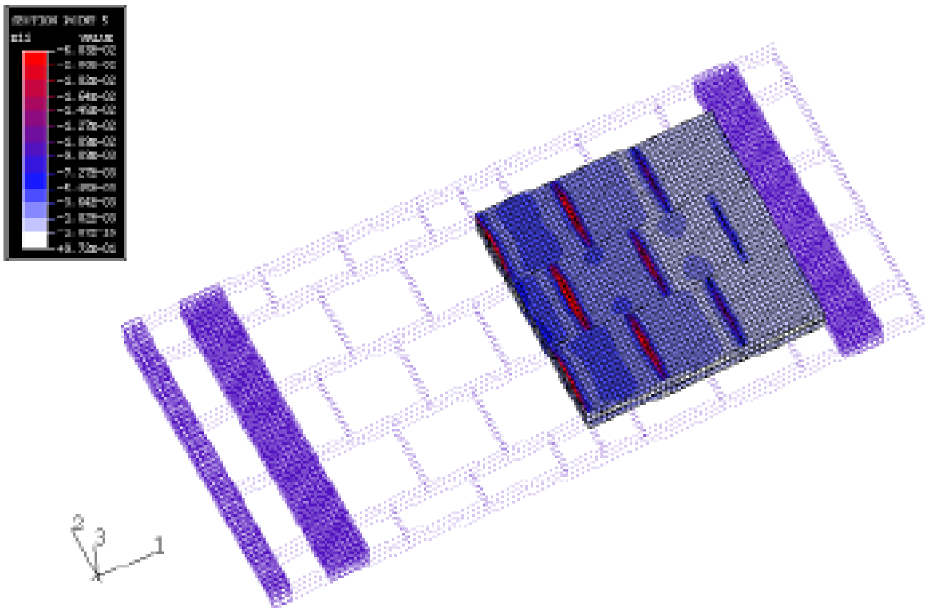


Figure 9. L4 contour plot of surface strains on selected superelements in the model. The superelement library used is L4.N8.

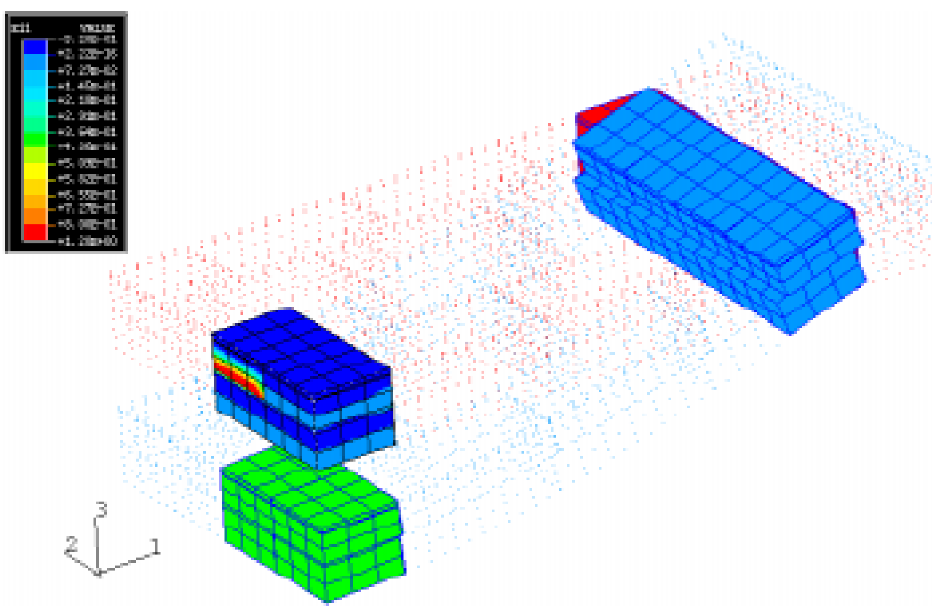


Figure 10. Superelement analysis based on library 3D.N3. Level-0 deformed plot shows retained nodes and standard elements above the support location (right of figure). Lower-level plots show strains and deformations inside selected superelements.